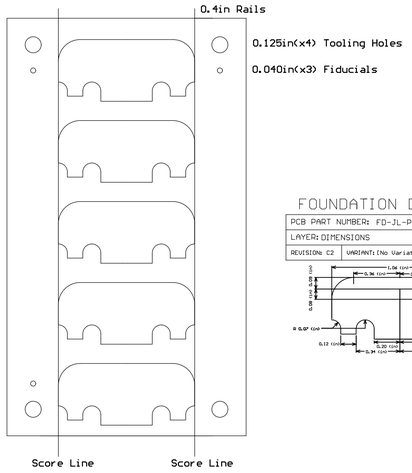
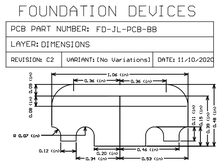
REVISIONS			
REV	DESCRIPTION	DATE	APPROVED
C1	Release for Production	07/29/2020	M. Beach
C2	Updated panalization drawing and notes.	11/09/2020	M. Beach

NOTES: (UNLESS OTHERWISE SPECIFIED)

1. THIS DRAWING SPECIFIES THE REQUIREMENTS FOR A PRINTED CIRCUIT BOARD IN ACCORDANCE WITH SPECIFICATION IPC-6012 CLASS 2 (LATEST REVISION).
2. THE PCB MUST BE LEAD FREE ASSEMBLY PROCESS COMPATIBLE AND MUST BE ABLE TO HANDLE A MINIMUM OF 5 CYCLES AT 260 DEGREES CELSIUS FOR 10 SECONDS.
3. BASE MATERIAL - LAMINATE AND PREPREG SHALL MEET IPC-4101E/124 or /129
   TG - MUST BE GREATER THAN OR EQUAL TO 150 DEGREES CELSIUS.
   TD - MUST BE GREATER THAN OR EQUAL TO 325 DEGREES CELSIUS.
4. COPPER FOIL WEIGHT - SEE STACKUP DETAIL 'A'
5. MINIMUM CONDUCTIVE WIDTH/SPACING TO BE 0.2mm/0.2mm
6. PLATING FINISH: ENIG
7. SOLDERMASK - TO MEET THE REQUIRMENTS OF IPC-SM-840E (OR LATEST REVISION).
   - BLACK COLOR. BOTH SIDES.
   - SOLDERMASK ON SOLDERABLE SURFACES IS PROHIBITED.
   - ALL CIRCUITRY DEFINED AS COVERED SHALL NOT BE EXPOSED.
8. SILKSCREEN - WHITE EPOXY OR ACRYLIC INK. BOTH SIDES. NO SILKSCREEN ON ANY EXPOSED COPPER FEATURE.
9. ELECTRICAL TEST - 100% IPCD356.
10. DFM CHECK MUST BE RUN ON BOARD DATA BEFORE BUILDING BOARDS. UNLESS PRIOR APPROVAL IS GIVEN IN WRITING BY FOUNDATION.
11. FINISHED PCB THICKNESS IS CRITICAL: FROM L1 PAD TO L2 PAD TO BE 1.6mm +/- 10%
12. TWO SOLDER SAMPLES TO BE PROVIDED
13. SUPPLIER MARKINGS - ON SECONDARY SIDE ONLY.
    - MUST BE UL RECOGNIZED AND MUST HAVE AN ID THAT CONFORMS TO UL94V-0
14. FINISHED PCB MUST BE PANELIZED FOR ASSEMBLY ACCORDING TO CONTRACT MANUFACTURER'S REQUIREMENTS.
    - SEE 'DETAIL B' FOR PREFERED PANEL LAYOUT.
    - MANUFACTURER CAN PROPOSE PREFERED PANAL DIMENSIONS AND BOARD COUNT. PANEL MUST BE APPROVED BY FOUNDATION DEVICES.
    - LEAVE AT LEAST 3.2mm GAP BETWEEN PCB'S ON PANEL.
    - ROUT ENTIRE BOARD OUTLINE EXCEPT WHEN OVERLAPPING THE VERTICAL SCORE LINES
    - PANELIZATION MUST BE APPROVED BY CONTRACT MANUFACTURER.

Panelization (Detail B)





Layer Stack-Up (Detail A)

Layer Name	Gerber Document	Copper Thickness	Insulator Thickness
Top Solder Mask	(.GTS)		
L1 0.5oz Plated Copper Foil	(.GTL)	0.03mm	1.540mm
L2 0.5oz Plated Copper Foil	(.GBL)	0.03mm	
Bottom Solder Mask	(.GBS)		
Total Board Thickness:		1.60mm	

LICENCE: CERN-OHL-S v2

SPDX-FileCopyrightText: 2020 Foundation Devices, Inc. <hello@foundationdevices.com>
SPDX-License-Identifier: CERN-OHL-S-2.0

You may redistribute and modify this source and make products using it under the terms of the CERN-OHL-S v2 (https://ohwr.org/cern_ohl_s_v2.txt).

This source is distributed WITHOUT ANY EXPRESS OR IMPLIED WARRANTY, INCLUDING OF MERCHANTABILITY, SATISFACTORY QUALITY AND FITNESS FOR A PARTICULAR PURPOSE. Please see the CERN-OHL-S v2 for applicable conditions.

Source location: foundationdevices.com/passport-electronics

As per CERN-OHL-S v2 section 4, should You produce hardware based on this source, You must where practicable and applicable maintain the Source Location visible (1) on the packaging of the hardware, (2) on the circuit board(s) via silkscreen or copper, (3) in any documentation, and (4) on other products you make using this source.

DRAWN BY M. Beach	FOUNDATION		Foundation Devices Inc. 6 Liberty Square #6018 Boston, MA 02109	
APPROVED BY M. Beach	PCB PART NUMBER: FD-JL-PCB-BB			
APPROVED ON 11/10/2020	SIZE C	DATE: 11/10/2020	VARIANT: [No Variations]	REV C2
	SCALE: 1/1	DO NOT SCALE DRAWING	SHEET 1 OF 1	